DOCUMENT RESUME

ED 182 572 CE 024 257

AUTHOR Selleck, Ben; Espy, John
TITLE Nuclear Technology. Course 27: Metrology. Module 27-5, Tolerancing.
INSTITUTION General Physics Corp., Columbia, Md.
SPONS AGENCY Office of Education (DHEW), Washington, D.C.
PUB DATE 79
CONTRACT 300-77-0340
NOTE 60p.; Small, light type in figures will not reproduce well. For related documents see CE 024 245-298.

EDRS PRICE MF01/PC03 Plus Postage.
DESCRIPTORS Course Content; Energy; Learning Activities; Learning Modules; *Measurement; *Measurement Instruments; Measurement Techniques; *Nuclear Physics; Postsecondary Education; *Quality Control; *Standards; Teaching Guides; *Technical Education
IDENTIFIERS Nuclear Energy; Nuclear Energy Occupations; *Nuclear Power Plant Technicians; *Nuclear Technology; Tolerance

ABSTRACT

This fifth in a series of eight modules for a course titled Metrology describes the application of the American National Standard (ANSI Y14.5-1973) for dimensioning and tolerancing and gives guidance on interpreting form and location controls consistent with the national standard. The module follows a typical format that includes the following sections: (1) introduction, (2) module prerequisites, (3) objectives, (4) notes to instructor/student, (5) subject matter, (6) materials needed, (7) procedures, (8) data table, (9) reference materials, (10) test. (The course is one of fifteen technical courses for the second year of a two-year postsecondary curriculum in Nuclear Quality Assurance/Quality Control Technology.) (LRA)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

ED182572

NUCLEAR TECHNOLOGY

U.S. DEPARTMENT OF HEALTH, EDUCATION & WELFARE
NATIONAL INSTITUTE OF EDUCATION

THIS DOCUMENT HAS BEEN REPRODUCED EXACTLY AS RECEIVED FROM THE PERSON OR ORGANIZATION ORIGINATING IT. POINTS OF VIEW OR OPINIONS STATED DO NOT NECESSARILY REPRESENT OFFICIAL NATIONAL INSTITUTE OF EDUCATION POSITION OR POLICY

COURSE 27 METROLOGY

MODULE 27-5 Tolerancing

CE 024 257

GENERAL PHYSICS CORPORATION
Columbia , Maryland

COURSE MODULES

COURSE 27 METROLOGY

Module 27-1 Fundamentals of Metrology

Module 27-2 Fixed Gages, Dividers, Calipers and Micrometers

Module 27-3 Gage Blocks, Mechanical Comparators and Electronic Comparators

Module 27-4 Angle Measurement Instruments, Optical Projectors and Surface Texture Gages

Module 27-5 Tolerancing

Module 27-6 Calibration

Module 27-7 Statistical Techniques in Metrology

Module 27-8 The Total Measurement System

The material in this module was developed by General Physics Corporation under a contract with the United States Department of Health, Education and Welfare, Office of Education Contract Number 300770340. Contractors undertaking such projects are encouraged to express freely their professional judgment in the conduct of the project. Therefore, points of view or opinions stated do not necessarily represent official Office of Education position or policy.

This module was prepared for General Physics Corporation by Ben Selleck, Ben-Sel Associates, and John Espy, General Physics Corporation

GENERAL PHYSICS CORPORATION

1979

INTRODUCTION

The sizes of individual features normally give insufficient information to produce a part as envisioned by the designer. In addition to size of a feature, the form of the feature can be of importance, e.g., straightness, flatness, roundness, and profile. Also, the form of the feature with respect to other features can be of importance, e.g., angularity, perpendicularity, parallelism, and runout. In addition, the location of the feature with respect to other features can be of importance, e.g., position, concentricity, and symmetry. The axiom "you can't fit a square peg in a round hole" basically summarizes the need for form and location control.

The American National Standard ANSI Y14.5-1973, Dimensioning and Tolerancing, is used throughout the nuclear industry as the standard method of applying form and location controls on drawings and interpreting the meaning of such controls. This module describes the application of ANSI Y14.5-1973 and gives guidance on interpreting form and location controls consistent with the national standard. In this regard, this module quotes three rules from ANSI Y14.5-1973 which serve as the basis of drawing interpretation with respect to form and location. The applicability of these rules is subsequently given as each characteristic of form and location is introduced.

The terminology introduced by this module is defined in Appendix A to the module.

The student/trainee should refer to the Appendix and to the three rules as necessary in studying the application of each characteristic of form and location.

MODULE PREREQUISITES

The student/trainee should be familiar with blueprint reading and should have completed courses in algebra and plane geometry. The student/trainee shall also have completed all previous modules of this course.

OBJECTIVES

Upon completion of this module, the student/trainee shall be able to do the following:

1. Define the basic terms of maximum material condition, least material condition, regardless of feature size, straightness, flatness, roundness, cylindricity, perpendicularity, parallelism, true position, concentricity, and runout.
2. Correctly interpret drawing requirements for size of individual features including bilateral, unilateral, and limit tolerancing.
3. Demonstrate application of rounding principles by correctly rounding given values.
4. Match geometric terms with their symbols.
5. Demonstrate the ability to interpret drawings by describing the applicable rule of tolerancing for a given geometric characteristic.
6. Demonstrate the ability to interpret drawings by measuring geometric characteristics of parts and comparing measured values with stated tolerances.

NOTES TO INSTRUCTOR / STUDENT

Laboratory exercises are recommended to develop student/trainee skills in making measurements of form and location. Measuring devices which will be necessary are those used for Modules 27-2, 27-3, and 27-4. Advance preparation will be necessary to provide for a selection of parts and instruments to be used for measurement of:

- Straightness
- Flatness
- Roundness
- Cylindricity
- Angularity
- Perpendicularity
- Parallelism
- Position
- Concentricity
- Symmetry
- Runout

SUBJECT MATTER

DEFINITIONS OF TERMS

Definitions of terms introduced by this module are given in Appendix A along with symbols representing such terms. The student/trainee should refer to Appendix A as each new term is introduced by the text. In addition, the definitions and symbols of the following terms should be memorized at the point of their introduction in the text:

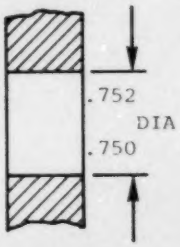
- Maximum Material Condition (MMC)
- Least Material Condition (LMC)
- Regardless of Feature Size (RFS)
- Straightness
- Flatness
- Roundness
- Cylindricity
- Perpendicularity
- Parallelism
- True Position
- Concentricity
- Runout

GENERAL TOLERANCING

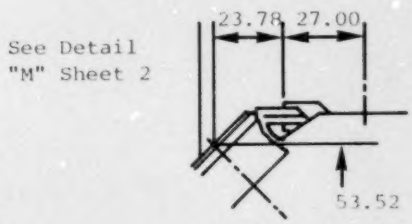
Review of Engineering Drawing Applications

The dimensional inspector and QA/QC technician must correctly interpret engineering drawings in order to properly perform the inspection function. Tolerances may be expressed on drawings by any combination of the following:

- As specific limits or a tolerance applied directly to the dimension:



- In the form of a special note referring to specific dimensions:
  - NOTE: The 3.50 and 1.750 diameters apply after chrome plating.
- As specified in other documents referenced on the drawing for specific features:



- In a general tolerance note referring to all dimensions on the drawing for which tolerances are not otherwise specified:

Tolerance Unless Otherwise Specified							
	Up To 6.00	Over 6.00 to 12.00	Over 12.00 to 24.00	Over 24.00 to 60.00	Over 60.00 to 120.00	Over 120.00 to 360.00	Over 360.00
All Dim.	± .12	± .18	± .26	± .38	$^{+.50}_{-.38}$	$^{+.76}_{-.50}$	$^{+1.26}_{-.76}$
Angular Tolerance ± 1^{o} Except Weld Geometry							
Weld Geometry $+2^{o}$ -0^{o}							

In view of these four applications, persons responsible for the inspection function should develop a habitual practice of reviewing drawings prior to commencing an inspection activity. Such habitual review will tend to avoid misinterpretation or omission of drawing requirements. A recommended sequence for such review is:

- Review all drawing notes to determine those that apply to the inspection function.
- Review the general tolerancing note for familiarity with applicable tolerances where no other tolerancing methods are used.
- Review the specific dimensions to be inspected to determine the specific tolerancing method applicable to such dimensions.

Review of Tolerance Expression

Engineering drawings express dimensions and limits or tolerances with the same number of decimal places. For example:

Correct	Incorrect
0.870-0.875	0.87-0.875
0.500 $\pm$ 0.005	0.5 $\pm$ 0.005

The same expression is used in the inspection function when recording measured values. That is, the measured value is recorded to at least the same number of decimals as specified by the drawing.

Review of Dimensional Tolerancing Methods

Drawings specify the acceptable range of dimensions in one or two methods as follows:

- Limit Dimensional Method - The high limit (maximum value) is placed above the low limit (minimum value) or when expressed in a single line the low limit precedes the high limit and both are separated by a dash. Examples are:

$$\begin{matrix} 0.750 \\ 0.748 \end{matrix} \qquad 0.748 - 0.750$$

- Plus and Minus Tolerancing Method - The dimension is given first and is followed by a plus and minus expression of tolerance. The plus and minus expression of tolerance may be unilaterial or bilateral.

Unilateral	Bilateral
$2.500 \pm \begin{matrix} 0.000 \\ 0.005 \end{matrix}$	6.250 ± 0.005
$1..25 \pm \begin{matrix} 0.003 \\ 0.000 \end{matrix}$	0.750 ± 0.001

Interpretation of Limits

All limits are absolute. Regardless of the number of decimal places, the limit is treated as if it were followed by an infinite number of zeroes. To determine conformance within limits, the measured value is compared directly with the limit and any deviation outside the limit signifies nonconformance.

The rule of any deviation signifying nonconformance is to be followed at all times by the inspection function. No matter how small the deviation outside the limit, the inspection function is not authorized to accept the deviation. Instead, an engineering review is necessary to determine the effect of the deviation on the fit, function, and interchangeability of the item.

There are special cases where the limits are waived because of engineering considerations. Such cases are documented to show that fit, function and interchangeability are not adversely affected. Such engineering decisions are not to be interpreted as negating the importance of the inspection activity nor indicating that a deviation is acceptable the next time it is detected. Each deviation must be evaluated on its own merits. If repetitive deviations are acceptable, engineering should change the limits such that the inspection function will not find measured values within the specified limits.

The above rule also brings to bear important concepts with respect to precision and as will be described later, the rounding of measured values. As described in Module 27-1, the element of precision that is termed discrimination becomes most important as limits are approached. In general the 10/1 or 4/1 rule described in Module 27-1 provides sufficient discrimination to permit directly comparing measured values with limits. But what happens when the measured value appears to be at the limit? The best rule to follow is to select a more discriminating inspection device that will measure to an additional decimal place. For example, a vernier micrometer reading of 2.5000 for a $2.5000\ ^{+0.0000}_{-0.0005}$ dimension should result in a "referee" inspection with a device (e.g., test indicator) that has discrimination finer than 0.0001.

Rounding of Measured Values

Rounding of measured values is desirable to allow recording of inspection results to the same number of decimals places as the drawing requirement. When a number is to be rounded to fewer decimal places the following procedure applies:

- When the first digit discarded is less than 5, the last digit retained must not be changed. For example 3.46325 rounded to three decimal places is 3.463; the same value rounded to two decimal places 3.46.
- When the first digit discarded is greater than 5, or if it is a 5 followed by at least one digit other than 0, the last figure retained must be increased by one unit. For example: 8.37652 rounded to three decimal places is 8.377; the same value rounded to two decimal places is 8.38.
- When the first digit discarded is exactly 5, followed only by zeroes, the last digit retained must be rounded upward if it is an odd number, but not changed if it is an even number. For example: 4.365 rounded to two decimal places becomes 4.36; and 4.355 would also round to the same value, 4.36, if rounded to two decimal places.
- Measured values are rounded directly to the nearest value having the desired number of decimal places, rounding is not done in successive steps to less places. For example: 27.46 rounded to a whole number is 27. This is correct because the ".46" is less than one-half. 27.46 rounded to one decimal place is 27.5. But, if 27.5 is in turn rounded to a whole number, this is successive rounding and the result, 28, is incorrect.
- Rounding is accomplished after completion of calculations. In the fcllowing example of comparative measurements, premature rounding erroneously shows nonconformance:

Limit	Calculations	Premature Rounding
0.7505 Maximum	1.36435 (Feature)	1.3644
	-0.61385 (Datum)	-0.6138
	0.75050 (Result)	0.7506

- Rounding is not permitted to change a measured value from a nonconforming to a conforming state. For example, if 0.2251 is measured for a 0.225 maximum limit, the value of 0.2251 is reported and the item is considered as nonconforming.

FORM CONTROL

Affect of Size Features on Form

Engineering drawings illustrate size features as if they have perfect geometric form. However, the size limits or tolerances permit the feature to deviate from perfect geometric form. This is illustrated in Figure 27-5-1.

As indicated by Figure 27-5-1, limits of size without limits of geometric form permit variations in geometric form as long as the size limits are met. This means that the form of physical boundaries of the size feature can vary along the length of the boundaries provided much variation does not result in exceeding the size limits.

Unless otherwise specified, the size limits of an individual feature prescribe the extent within which variations of geometric form as well as size area allowed. This principle is illustrated by Figure 27-5-1 and it is expressed by Rule #1 as given by ASNI Y14.5-1973:

- Rule #1 - Unless otherwise specified, the limits of the dimension of an individual feature as well as its size. This control is defined as:

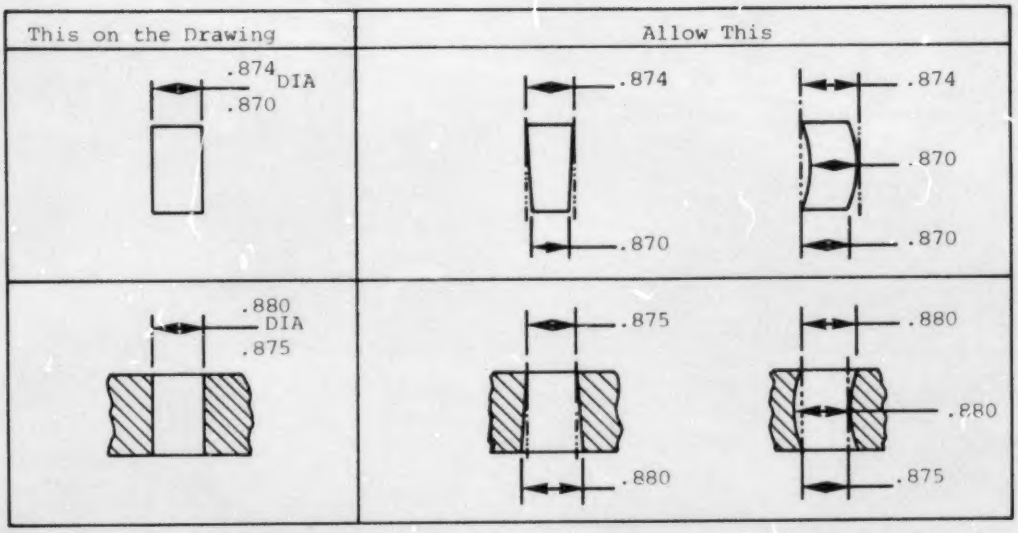


Figure 27-5-1 Extreme Variations of Form Allowed by a Size Tolerance

- No element of the actual feature shall be extend beyond a boundary of perfect form at Maximum Material Condition (MMC). (Except for material stock such as bars, sheets, and tubing where established standards specify such forms as straightness, flatness and others.) For the pin in Figure 27-5-1, this means that no portion of the pins outer surface can lie outside the boundary of a perfect cylinder whose diameter is 0.874.
- The actual size of the feature at any cross-section shall be within the Least Material Condition (LMC) limit. For the pin in Figure 27-5-1, this means that no measurement of pin diameter can be less than 0.870.

EXAMPLE: APPLICATION OF RULE #1

Given: The 0.875-0.880 diameter through hole illustrated in Figure 27-5-1.

Problem: If all diametrical points measure 0.875, what must be the form of the cylindrical boundaries of the hole?

Solution: Since the diameter at all points equals the limit at MMC, perfect cylindrical form must exist to avoid the extension of at least one element of the feature beyond a boundary of perfect form at MMC.

Problem: If the diameter measures 0.881 at one set of diametrical points but measures between 0.875 and 0.880 at all other diameters, is the feature acceptable?

Solution: No, because one cross-section of the feature lines outside the limit at LMC.

Effects of RFS

Where a position or form tolerance is specified on an RFS basis, the specified tolerance is independent of the size of the considered feature. The position or form tolerance is limited to the specified value regardless of the actual size of the feature. Likewise, referencing a datum feature on an RFS basis means that a centering about its axis or center plane is necessary, irrespective of the actual size of the feature.

Effect of MMC

Where a tolerance of position or form is applied on an MCC basis, the specified tolerance is interdependent on the size of the considered feature. The tolerance is limited to the specified value if the feature is produced at its MCC limit of size. Where the actual size of the feature departed from MCC (in an acceptable direction), an increase in the tolerance is allowed equal to the amount of such

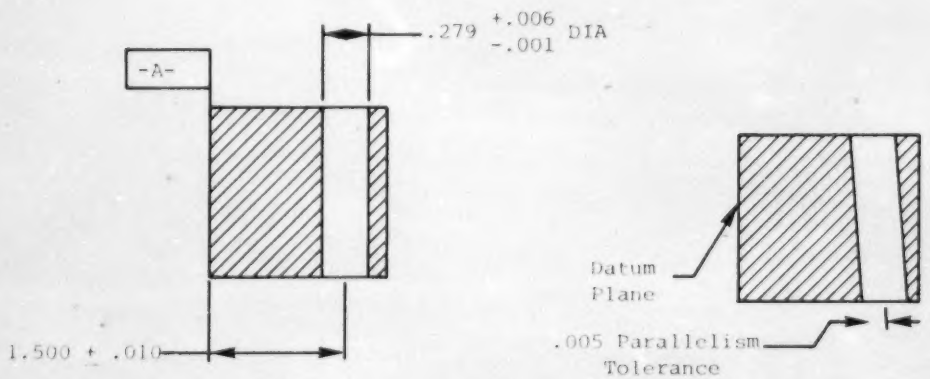
departure. The total permissible variation in position or form is maximum when the features is at LMC. Likewise, referencing a datum feature on an MCC basis means the datum is the axis or center plane of a diameter or width fixed at the MMC limit of size of the feature. Where the actual size of the datum feature has departed from MMC, a deviation is allowed between its axis or centerplane and the datum axis or center plane.

Effect of Zero Tolerance at MCC

Where a tolerance of position or form is applied on a zero tolerance at MMC basis, the tolerance is totally dependent on the size of the considered feature. No tolerance is allowed if the feature is at its MMC limit of size; and in such case, it must be located at true position or be perfect in form, as applicable. Where the actual size of the feature has departed from MMC, a tolerance is allowed equal to the amount of such departure. The total permissible variation in position or form is maximum when the feature is at LMC, unless a maximum is specified.

EXAMPLE: RFS AND MCC TOLERANCING

Given: The part illustrated by the following drawing:



16

Problem: If the drawing specified 0.005 parallelism of the 0.279 diameter to datum A on an RFS basis, what effect would the actual size of the diameter have on the parallelism tolerance?

Solution: None. The tolerance applies regardless of feature size by definition. Irrespective of the actual size of the 0.279 diameter, the parallelism to datum A must be within a 0.005 tolerance zone which is illustrated as follows in an exaggerated manner:

Actual Feature Size	Parallelism Tolerance Allowed
0.278	0.005
0.280	0.007
0.282	0.009
0.285	0.012

Problem: If the drawing specified 0.005 parallelism of the 0.279 diameter to datum A on an MCC basis, what effect would the actual size of the diameter have on the parallelism tolerance?

Solution: The diameter's departure from 0.278 (which is MCC) is added to the 0.005 tolerance with the maximum addition equal to 0.007 at LMC. Such effect is illustrated by the following table:

ACTUAL FEATURE SIZE	PARALLELISM TOLERANCE ALLOWED
0.278	0.005
0.280	0.007
0.282	0.009
0.285	0.012

Problem: If the drawing specified the parallelism of the 0.279 diameter to datum A on a zero at MMC basis, what effect would the actual size of the diameter have on the parallelism tolerance?

Solution: At MMC (0.278 diameter), zero tolerance applies and the diameter must be perfectly parallel to datum A. As the actual size of the diameter approaches LMC, the deviation of the actual size from MMC equals the parallelism tolerance. The maximum tolerance is 0.007 which occurs when the diameter is at LMC (0.285).

Discussion: If one imagines a shaft inserted in the 0.279 diameter, the tightest fit will occur at MMC. Suppose that the engineer knows that the shaft center line will be fixed in a parallel position to datum A at assembly. If so, the parallelism tolerance is based on permitting assembly with both the shaft and the diameter at MMC. Dependent on the function of the assembly, the engineer selects either RFS or MMC tolerancing of the parallelism. If tightness of the fit is not particularly important, MMC would be selected because in effect it gives a bonus tolerance for manufacturing and therefore, less parts are likely to be rejected.

SYMBOLOGY

Symbols for Geometric Characteristics

The standard symbols for geometric characteristics as given by ANSI Y14.5-1973 are illustrated in Figure 27-5-2. These symbols are typically used on engineering drawings in lieu of spelling out the characteristics.

		Characteristic		Notes
Individual Features	Form Tolerances	Straightness	—	1
		Flatness	▱	1
		Roundness (Circularity)	○	
		Cylindricity	⌭	
Individual or Related Features		Profile of a Line	⌒	2
		Profile of a Surface	⌓	2
Related Features		Angularity	∠	
		Penpendicularity (Squareness)	⊥	
		Parallelism	//	3
	Location Tolerances	Position	⌖	
		Concentricity	◎	3, 7
		Symmetry	⌯	5
	Runout Tolerances	Circular	↗	4
		Total	↗	4, 6

Note: 1) The symbol ⏦ formerly denoted flatness.
The symbol ⌒ or — formerly denoted flatness and straightness.

2) Considered "related" features where datums are specified.

3) The symbol ⫽ and ◉ formerly denoted parallelism and concentricity, respectively.

4) The symbol ↗ without the qualifier "CIRCULAR" formerly denoted total runout.

5) Where symmetry applies, it is preferred that the position symbol be used.

6) "TOTAL" must be specified under the feature control symbol.

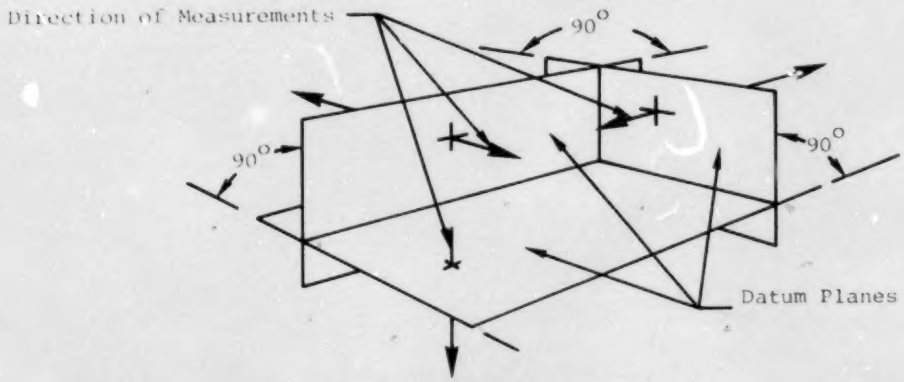
7) Consider the use of position or runout.

Where existing drawings using the above former symbols are continued in use, each former symbol denotes that geometric characteristic which is applicable to the specific type of feature shown.

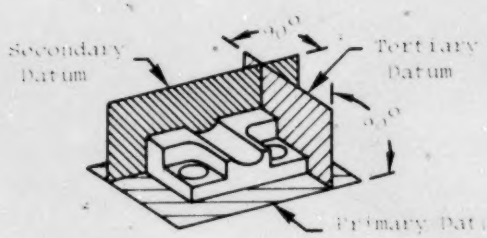
Figure 27-5-2 Geometric Characteristic Symbols

Datums and Their Symbols

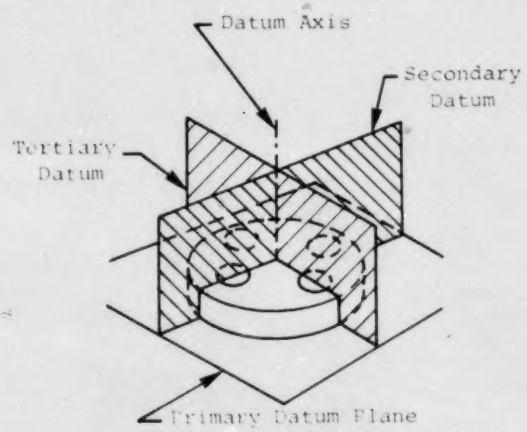
Engineering drawings specify surfaces or other features which are most important to the design, as origins (datums) for dimensions. Sufficient features are indicated to position the part in relation to a set of three mutually perpendicular planes as illustrated in Figure 27-5-3. All related measurements of the part originate from these planes.



(a) Datum Reference Planes



(b) Example of Flat Surface Datums



(c) Example of a Datums with Cylindrical Features

Figure 27-5-3 Reference Planes

Datums are identified in drawings by a boxed alpha designator as shown in Figure 27-5-4. Depending on the complexity of the part, the number of datums may range from one to several tens of datums. Each datum is given its own unique letter and all letters of the alphabet except I, O, and Q may be used. Where the single alpha series is exhausted, the double alpha series is used, i.e., AA through AZ.

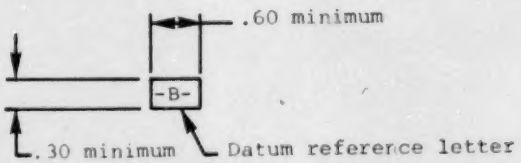


Figure 27-5-4 Datum Symbols and Size Thereof

Other Symbols

Other symbols commonly indicated on drawings are illustrated in Figure 27-5-5. Certain of these symbols are used with the feature control symbol which is described later.

Term	Abbreviation	Symbol
Maximum Material Condition	MMC	Ⓜ
Regardless of Future Size	RFS	Ⓢ
Diameter	DIA	⌀
Projected Tolerance Zone	Tol Zone Proj	Ⓟ
Reference	REF	(1.250)
Basic	BSC	3.875

Figure 27-5-5 Other Symbols

Feature Control Symbols

The feature control symbol indicates a position or form tolerance. It consists of a frame containing the geometric characteristic symbol (See Figure 27-5-2), datum references (if any), RFS or MMC modifier as appropriate, (See Rules #2 and #3), the diameter symbol where applicable, and the tolerance of the geometric characteristic. The feature control symbol illustrated in Figure 27-5-6 is read as follows: the angularity of the feature shall be within 0.005 on an MMC basis to datum C regardless of datum C size and also to datums B and A on an MMC datum B and A basis.

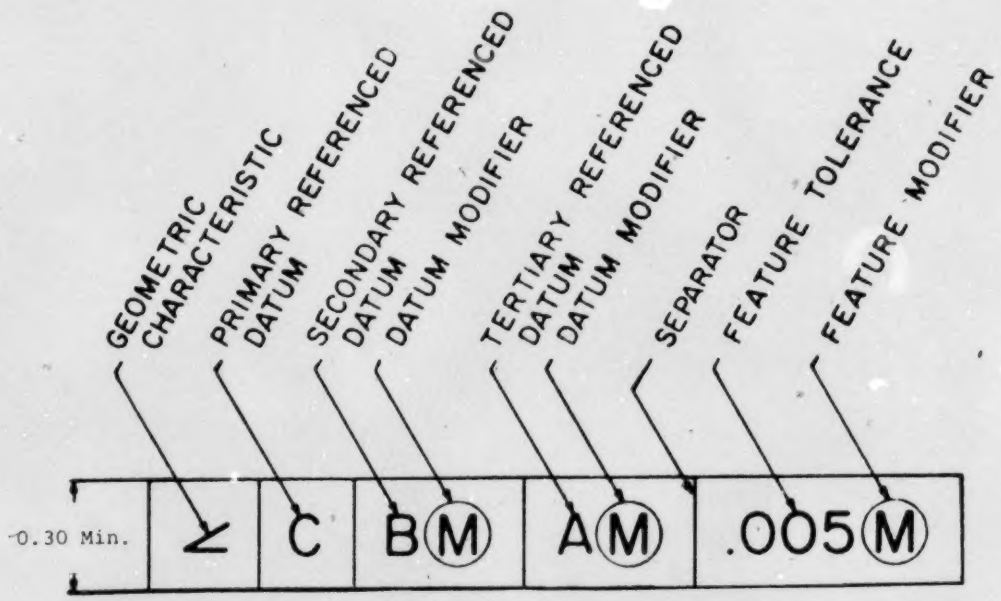


Figure 27-5-6 Illustration of Feature Control Symbol

FORM TOLERANCES

Form tolerances control straightness, flatness, roundness, cylindricity, profile, angularity, parallelism and perpendicularity (See Figure 27-5-2). Form tolerances are specified by the engineer where the size tolerances do not give the required control. Where the form tolerance value represents the diameter of a cylindrical zone, it is preceded in the feature control symbol by the diameter symbol.

Straightness

A straightness tolerance specifies a tolerance zone within which an axis or all points of the considered element must lie. It is applied in the drawing view where the elements to be controlled are represented by a straight line. Where straightness is specified independent of the individual size feature, Rule #1 also applies because one plane surface is the feature to be controlled. In such cases, a straightness tolerance smaller than the size tolerance serves to limit the variation in form. Figure 27-5-7 illustrates such a case. Per Rule #1, the size tolerance permits straightness of the pin in Figure 27-5-7 to vary from perfect straightness at MMC (0.615) to within 0.010 at LMC (0.605). However, the application of the straightness control feature in Figure 27-5-7 limits the maximum unstraightness of 0.003.

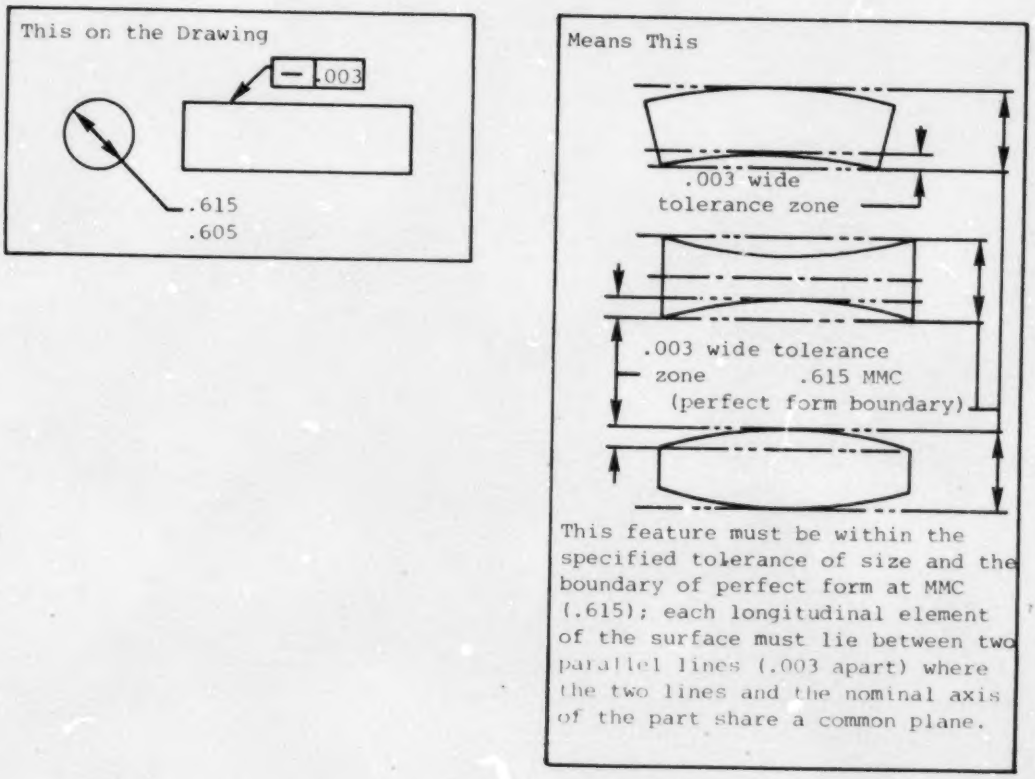

This on the Drawing
.003
.615
.605
Means This
.003 wide
tolerance zone
.003 wide tolerance
zone
.615 MMC
(perfect form boundary)
This feature must be within the specified tolerance of size and the boundary of perfect form at MMC (.615); each longitudinal element of the surface must lie between two parallel lines (.003 apart) where the two lines and the nominal axis of the part share a common plane.

Figure 27-5-7 Straightness Tolerance Independent of Size Tolerance

Where straightness is specified with the size dimension such that the feature axis is being controlled, Rule #1 no longer applies. Instead, the straightness tolerance is applied on an RFS or MMC basis in accordance with Rule #3.

Figure 27-5-8 illustrates straightness specified with the size dimension on an RFS basis. The pin in this figure must be straight within the straightness tolerance of 0.015 regardless of the actual size of the pin's diameter. Since the straightness tolerance in this case depicts a diametrical zone, the diameter symbol precedes the straightness tolerance.

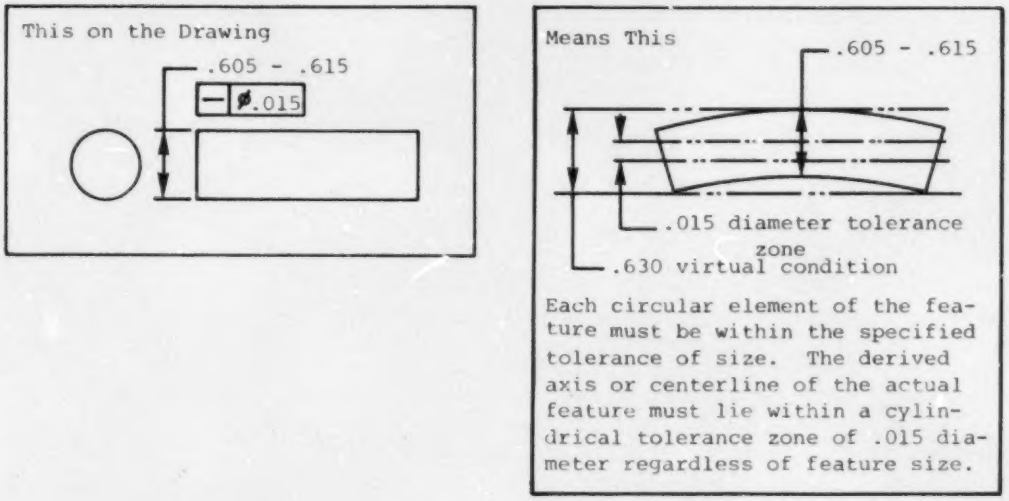

This on the Drawing
.605 - .615
⌀.015
Means This
.605 - .615
.015 diameter tolerance zone
.630 virtual condition
Each circular element of the feature must be within the specified tolerance of size. The derived axis or centerline of the actual feature must lie within a cylindrical tolerance zone of .015 diameter regardless of feature size.

Figure 27-5-8 Straightness Tolerance With Size Feature on RFS Basis

Figure 27-5-9 illustrates straightness specified with the size dimension on an MMC basis. The pin in this figure must meet the straightness tolerance when the pin's diameter is at MMC. However, the deviation from MMC toward LMC of the diameter is added to the straightness tolerance.

Measurement of straightness employs the comparison of the feature element with a line or plane of known straightness. Several typical techniques are illustrated in Table 27-5-1. (Please refer to the next page for Table 27-5-1.)

Flatness

A flatness tolerance specifies a tolerance zone defined by two parallel planes within which the surface must lie. The flatness form control symbol is attached to a leader directed to the surface or extension line thereof in the drawing view where the surface is represented by a line. Since flatness is associated with a one plant surface, RFS

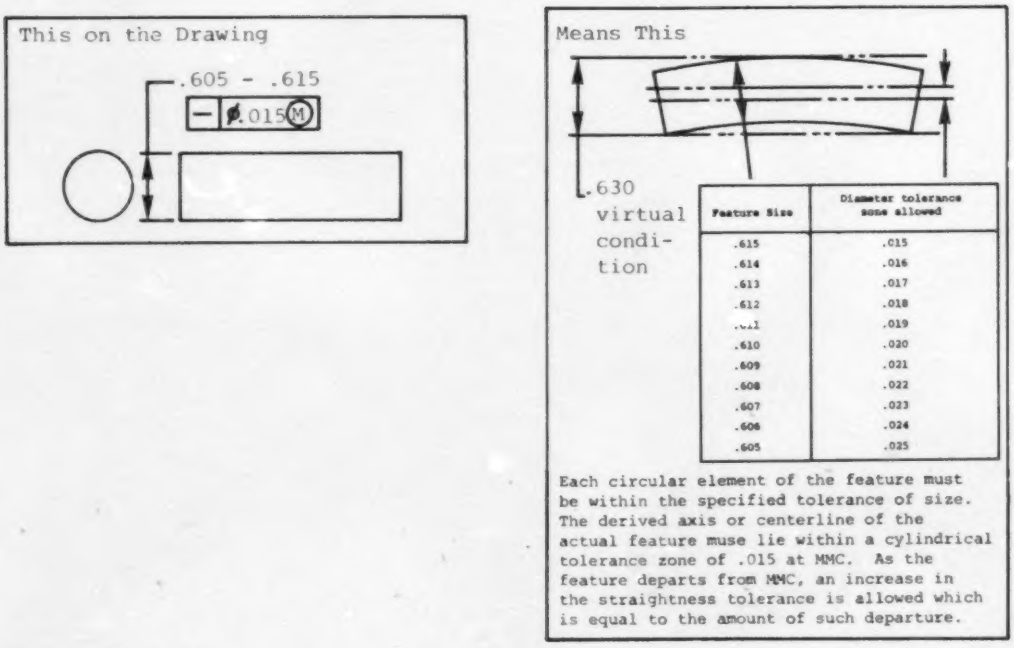

This on the Drawing
.605 - .615
⌀.015 M
Means This
.630
virtual
condi-
tion
Feature Size
Diameter tolerance zone allowed
.615 .015
.614 .016
.613 .017
.612 .018
.611 .019
.610 .020
.609 .021
.608 .022
.607 .023
.606 .024
.605 .025
Each circular element of the feature must be within the specified tolerance of size. The derived axis or centerline of the actual feature muse lie within a cylindrical tolerance zone of .015 at MMC. As the feature departs from MMC, an increase in the straightness tolerance is allowed which is equal to the amount of such departure.

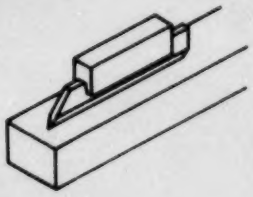
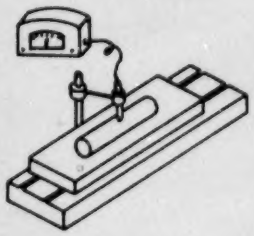
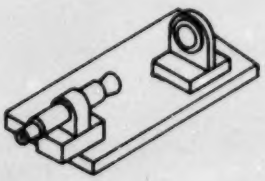
Figure 27-5-9 Straightness Tolerance With Size Feature on MMC Basis

and MMC are not applicable. Rule #1 and the flatness tolerance control flatness as indicated in Figure 27-5-10.

Typical measurement techniques for inspecting flatness are illustrated in Table 27-5-2. The height gage is suitable for use where the indicator discrimination is 1/4 or finer of the flatness tolerance. The autocollimator system has the capability of measuring the flatness of surface plates and other relatively large surfaces having flatness tolerances in microinches. The optical flat has the ability to measure flatness to within a few microinches. It is used on small surfaces having stringent flatness requirements such as the sensing surfaces of many measuring devices, (e.g., micrometer anvils).

Table 27-5-1

Straightness Inspections - Example Methods

Instruments	Diagram	Discussion
Knife edge rule		Toolmaker or reference quality knife edges having straightness accessories of about 0.00001 to 0.0001 are placed against the feature element. Ratio of contact of blueing or access by feeler gages provide inspection results.
Surface plate, vee blocks, height indicator gage		Part is staged on a surface plate which serves as the reference plane. Height indicator explores straightness of feature element. (NOTE: Similar inspection technique is use of bench centers to support the part from its axis.)
Precision slide and sensitive indicator		In this example, the height indicator is stationary and the part is moved with a precision slide. An electronic or mechanical test indicator measures variations in straightness. (A similar technique employs measuring machines whose indicator is mounted to precision slides.)
Optical telescope and optical targets		Optical targets moved along the feature element reflect variations in straightness as read by the optical telescope.

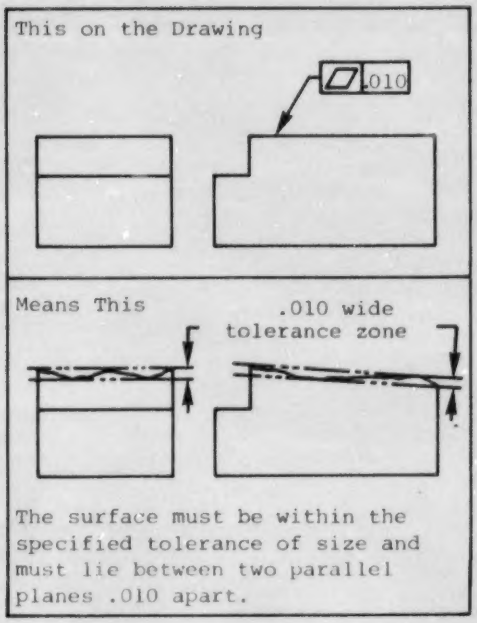
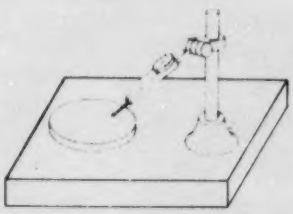
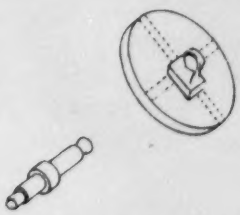
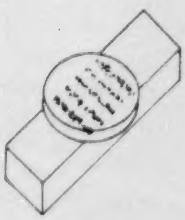

This on the Drawing
.010
Means This
.010 wide
tolerance zone
The surface must be within the specified tolerance of size and must lie between two parallel planes .010 apart.

Figure 27-5-10 Flatness Form Control

Roundness

A roundness tolerance can apply to either a cylindrical, conical, or spherical feature. It establishes a diametrical tolerance zone within which a selected circular surface of the feature must lie. Thus, roundness is sometimes termed circularity. As illustrated in Figure 27-5-11, Rule #1 and the roundness tolerance apply when roundness is specified by the drawing. Rule #1 applies because the surface of the feature is being controlled by roundness.

Table 27-5-2
Flatness Inspection - Example Methods

Instrument	Diagram	Discussion
An indicator stand and a surface plate		The surface to be measured is first set up to be parallel to the surface plate. The indicator measures variations caused by unflatness.
Autocollimator and reflector with base having appropriate pad separation		As the reflector is moved along various lines of position, unflatness tilts the reflector. The autocollimator has an eye piece for direct reading of graticles or has a photoelectric sensor.
Optical flat with monochromatic light or plano-interferometer		Monochromatic light of known wavelength is reflected by the air wedge between the optical flat and the work piece such that dark bands are visible. At perfect flatness the bands are straight and spaced exactly one-half wavelength. The extent of curvature of the bands corresponds to the degree of unflatness.

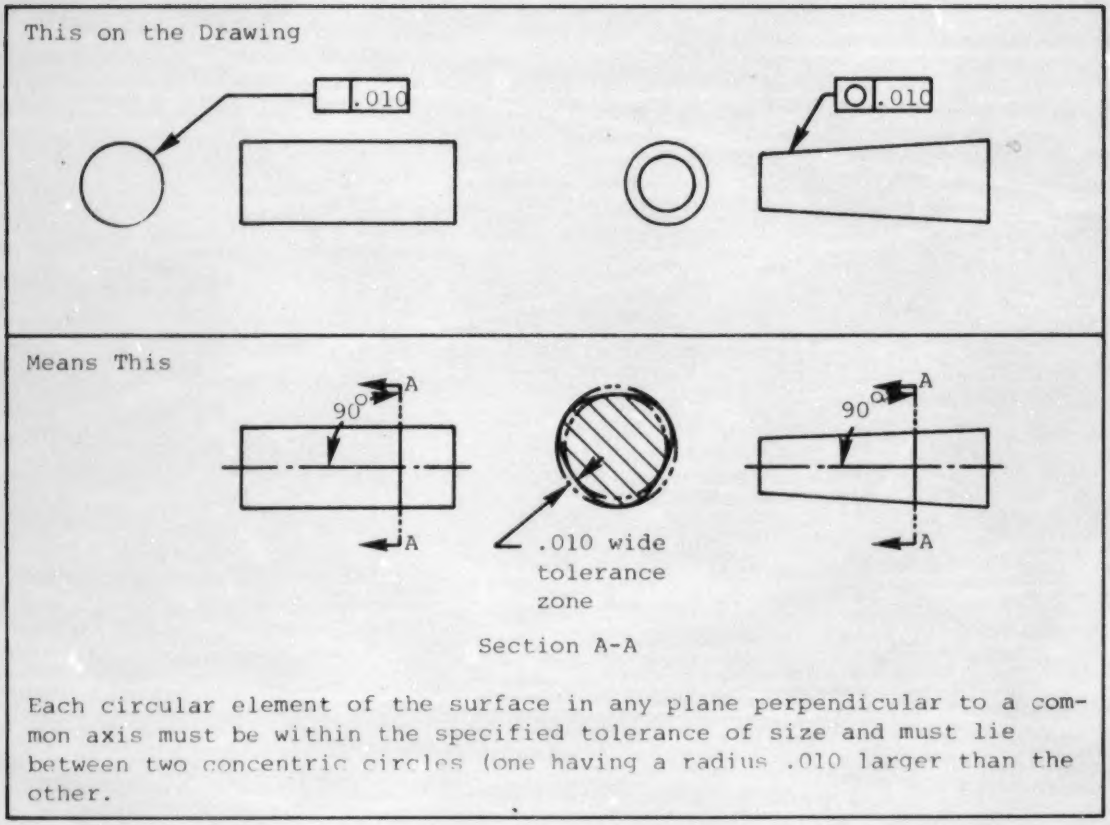
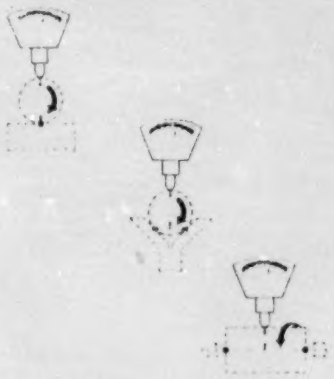
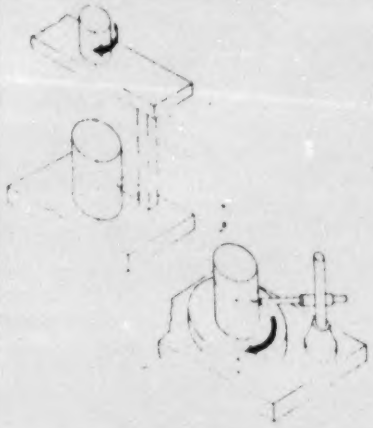


Figure 27-5-11 Roundness Form Control

Measurements of roundness are described in Table 27-5-3. The first type uses the feature as a reference datum. This type of roundness inspection compares diameters of the feature measured at a number of points. If permitted by the tolerance, standard inside and outside measuring devices, such as vernier calipers and micrometers, may also be used for this type of roundness inspection. Comparison of diameters for roundness inspection assumes that each diameter's center point falls on the same axis of the feature. It is also dependent on selecting both largest and smallest diameters within a radial plane. Therefore, comparison of diameters

Table 27-5-3
Roundness Inspection - Example Methods

Instrument	Diagram	Discussion
Height Gage with: a. reference plane b. vee block c. bench-center		With the reference plane and vee block techniques, variations in feature diameter are representative of variations in roundness. The vee block tends to yield larger indicator readings than the actual out of roundness and normally requires use of correction factors. The bench-center technique measures variations in radial height from the axis of the feature.
Electronic Profiling System with: a. rotating spindle b. rotating table		Profile systems are commonly used to plot variations of the feature to a precision axis of rotation provided by the spindle or table. Such systems also permit determination of actual contour and count of the number of periodic departure from roundness such as lobes.

to determine roundness is suitable when roundness and not profile is the major concern. When the profile is of major concern, (i.e., the distribution pattern of departures from roundness is of concern), a profiling system is used for measuring roundness.

Cylindricity

A cylindricity tolerance establishes a tolerance zone bounded by two concentric cylinders within which the feature surface must lie. Therefore, unlike roundness, cylindricity applies simultaneously to the entire surface. As illustrated in Figure 27-5-12, Rule #1 and the cylindrical tolerance apply to the feature. Rule #1 applies because the surface of the feature is being controlled by cylindricity.

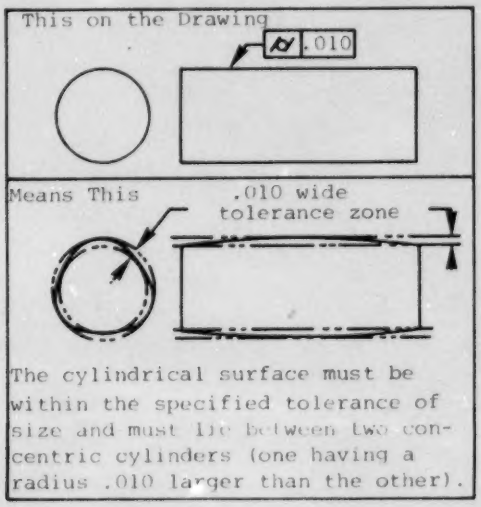

This on the Drawing
.010
Means This
.010 wide
tolerance zone
The cylindrical surface must be within the specified tolerance of size and must lie between two concentric cylinders (one having a radius .010 larger than the other).

Figure 27-5-12 Cylindricity Form Control

As described in Module 27-2, cylindricity can be measured on a GO-NOT GO basis by cylindrical plug and ring gages. In addition, the techniques of comparison of

diameters listed in Table 27-5-3 measure cylindricity when applied along the axis of the feature.

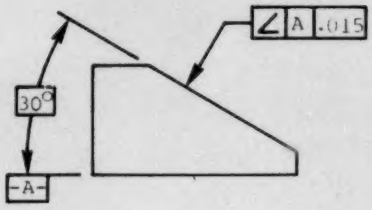
Angularity

Angularity tolerancing establishes a tolerance zone defined by two parallel planes at the specified basic angle within which (a) the surface of the considered feature must lie or (b) the axis of the considered feature must lie. When an angularity tolerance is applied to a surface which forms an angular boundary, Rule #1 and the angularity tolerance apply. When an angularity tolerance is applied to the axis of a feature, RFS or MMC applies in accordance with Rule #3, Figure 27-5-13 illustrates these concepts.

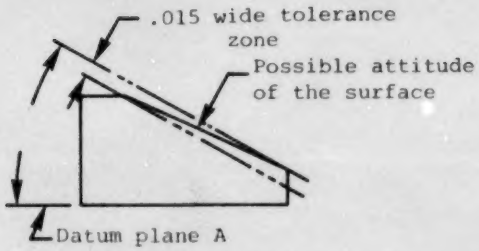
Perpendicularity

Perpendicularity can be applied to a surface, axis, or diametrical axis. In the first two applications, the perpendicularity tolerance zone is defined by two parallel planes perpendicular to a datum plane or axis within which the considered surface or axis must lie. When applied with a diameter feature symbol, the perpendicularity tolerance zone is defined by a cylindrical tolerance zone perpendicular to a datum plane within which the axis of the considered feature must lie. Whenever either application pertains to the perpendicularity of an axis, Rule #3 applies. When applied to a surface, Rule #3 applies. When applied to a surface, Rule #1 applies. These concepts are illustrated in Figure 27-5-14, 27-5-15, and 27-5-16.

a. This on the Drawing

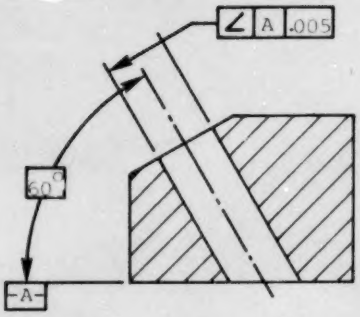


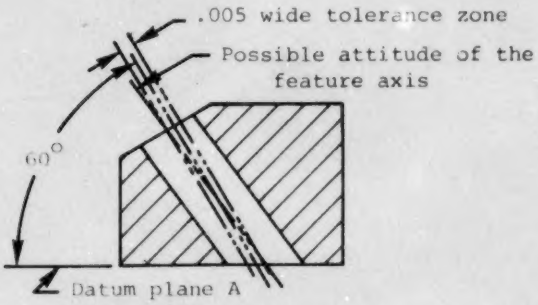
Means This



The surface must be within the specified tolerance of size and must lie between two parallel planes .015 apart which are inclined at 30° to the datum plane.

b.





The feature axis must be within the specified tolerance of location and must lie between two parallel planes .005 apart which are included at 60° to the datum plane.

(RFS)

Figure 27-5-13 Angularity Form Control

Angularity is inspected by the methods described by Module 27-4.

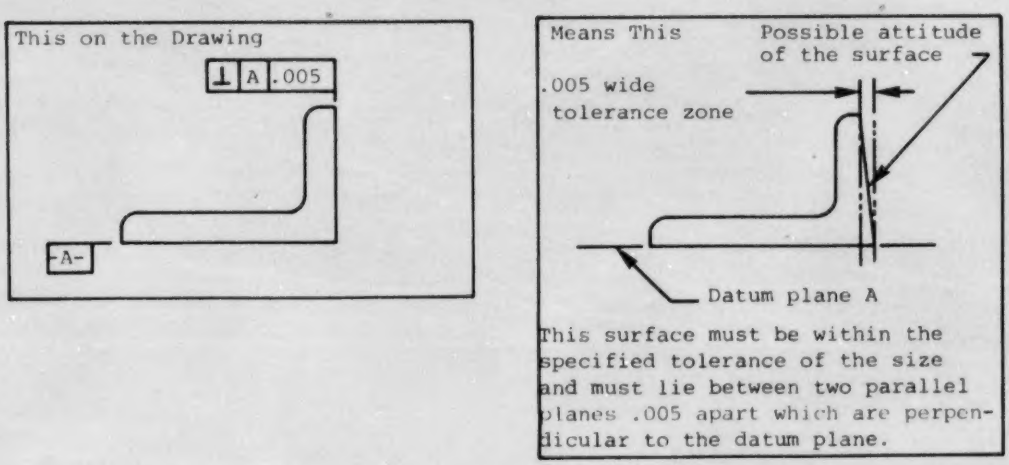

This on the Drawing
⊥ A .005
-A-
Means This
Possible attitude of the surface
.005 wide tolerance zone
Datum plane A
This surface must be within the specified tolerance of the size and must lie between two parallel planes .005 apart which are perpendicular to the datum plane.

Figure 27-5-14 Perpendicularity Control of a Surface (Rule #1)

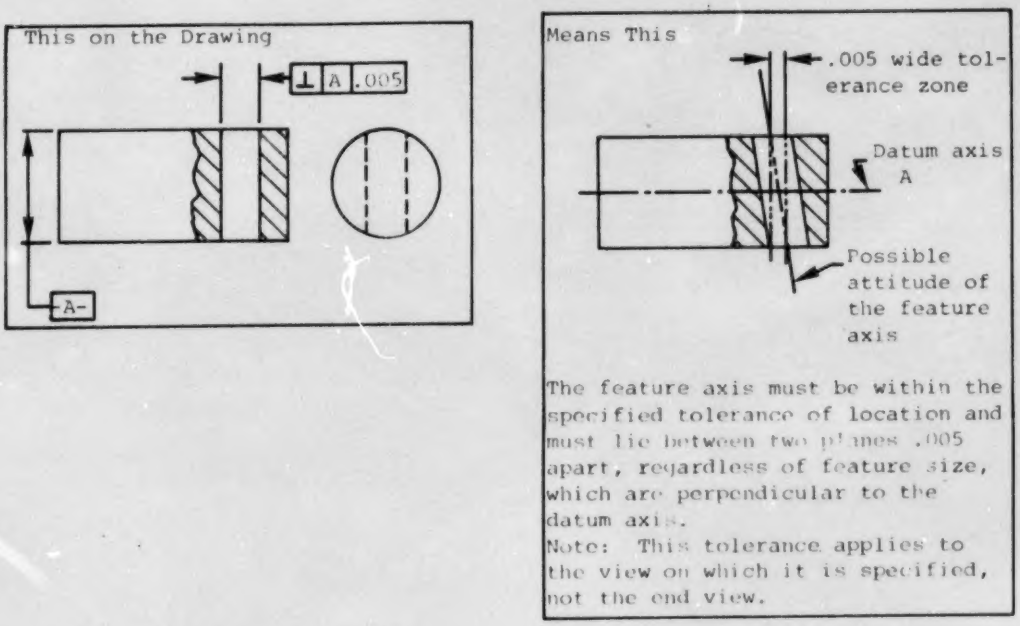

This on the Drawing
⊥ A .005
-A-
Means This
.005 wide tolerance zone
Datum axis A
Possible attitude of the feature axis
The feature axis must be within the specified tolerance of location and must lie between two planes .005 apart, regardless of feature size, which are perpendicular to the datum axis.
Note: This tolerance applies to the view on which it is specified, not the end view.

Figure 27-5-15 Perpendicularity Control of an Axis Without Diameter Symbol (Rule #3)

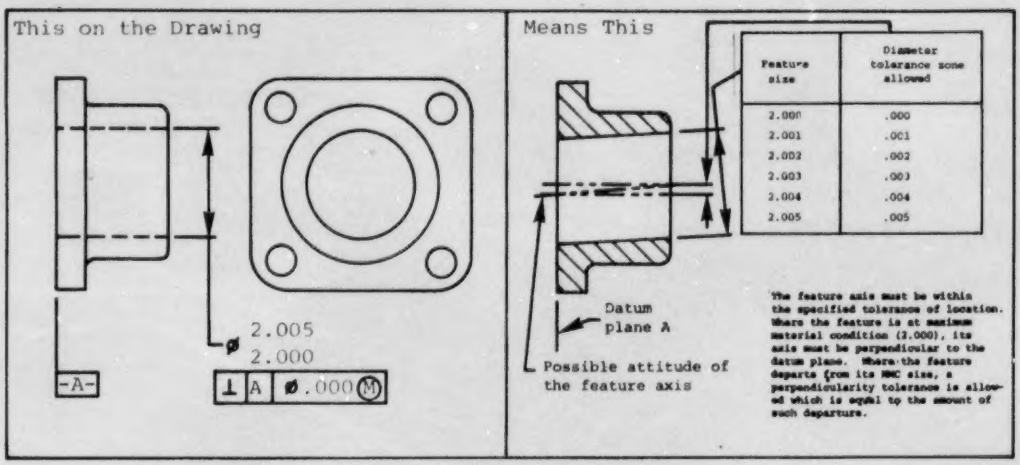

This on the Drawing
2.005
2.000
-A-
⊥ A ⌀.000 Ⓜ
Means This
Feature size
Diameter tolerance zone allowed
2.000 .000
2.001 .001
2.002 .002
2.003 .003
2.004 .004
2.005 .005
Datum plane A
Possible attitude of the feature axis
The feature axis must be within the specified tolerance of location. Where the feature is at maximum material condition (2.000), its axis must be perpendicular to the datum plane. Where the feature departs from its MMC size, a perpendicularity tolerance is allowed which is equal to the amount of such departure.

Figure 27-5-16 Perpendicularity Control With Diameter Symbol (Rule #3)

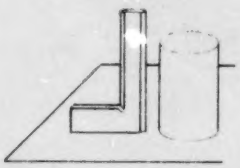
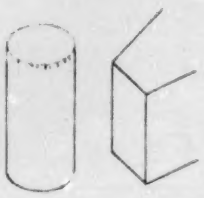
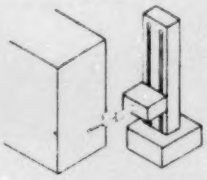
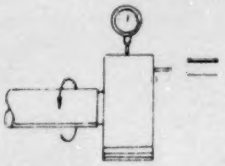
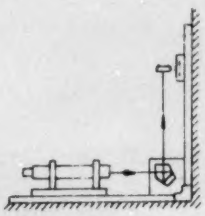
Example methods of measuring perpendicularity are given in Table 27-5-4 on the following page.

Parallelism

A tolerance of parallelism can be applied in the same manner as perpendicularity. When it applies to a surface or an axis with respect to a datum surface or axis, parallelism tolerance represents two planes or lines parallel to the datum within which the surface or axis of the considered feature must lie. When applied with the diameter feature symbol, parallelism tolerance represents a cylindrical tolerance zone parallel to the datum within which the axis of the considered feature must lie. When applied to a surface, Rule #1 also applies. When applied to an axis, RFS or MMC applies in accordance with Rule #3. These concepts are illustrated in Figures 27-5-17, 27-5-18, and 27-5-19.

Table 27-5-4

Perpendicularity Measurement - Example Methods

Instruments	Sketch	Discussion
Toolmaker square		Both the part and the tool rest on a surface plate. Gap between the edge of the square and sections along the contacted object surface is either visually observed or measured with a feeler gage.
Cylindrical square, with one face off angle and with graduations on the surface		The cylindrical square is brought into contact with the surface to be measured for perpendicularity. Curved graduation lines on the cylinder surface indicate the tilt angle of the part surface, by rotating the cylinder to an orientation producing the least light gap.
Height gage stand with traveling indicator head		A special height gage stand with guideways to provide straight and vertical guidance to a head which carries an indicator in contact with the object surface.
Indicators and supporting members for the part		Variations from the perpendicular between two surfaces which are related to a common axis can be measured by rotating the part around that axis and indicating the runout of selected circular paths.
Autocollimator optical square and reflector		Aligning an autocollimator with the datum plane and turning the emitted light at a right angle with an optical square, permits measurement of perpendicularity by observing the reflection from a mirror moved along the inter-related plane.

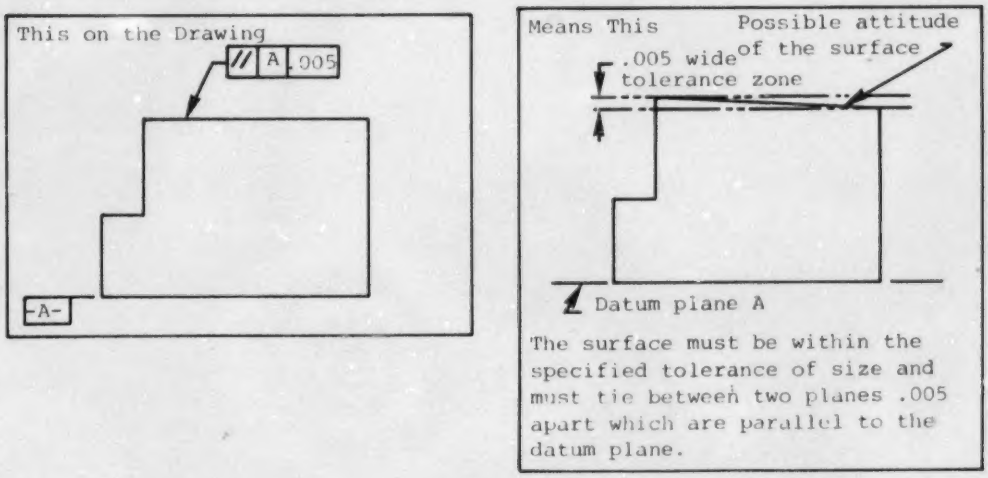

This on the Drawing
// A .005
-A-
Means This
Possible attitude of the surface
.005 wide tolerance zone
Datum plane A
The surface must be within the specified tolerance of size and must lie between two planes .005 apart which are parallel to the datum plane.

Figure 27-5-17 Parallelism Form Control of a Surface (Rule #1)

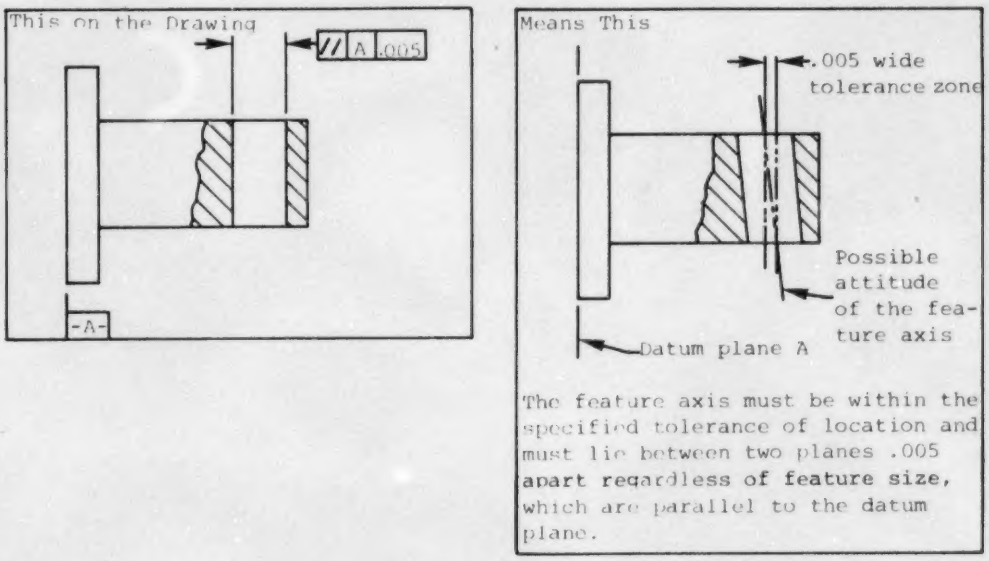

This on the Drawing
// A .005
-A-
Means This
.005 wide tolerance zone
Possible attitude of the feature axis
Datum plane A
The feature axis must be within the specified tolerance of location and must lie between two planes .005 apart regardless of feature size, which are parallel to the datum plane.

Figure 27-5-18 Parallelism Form Control of an Axis (Rule #3)

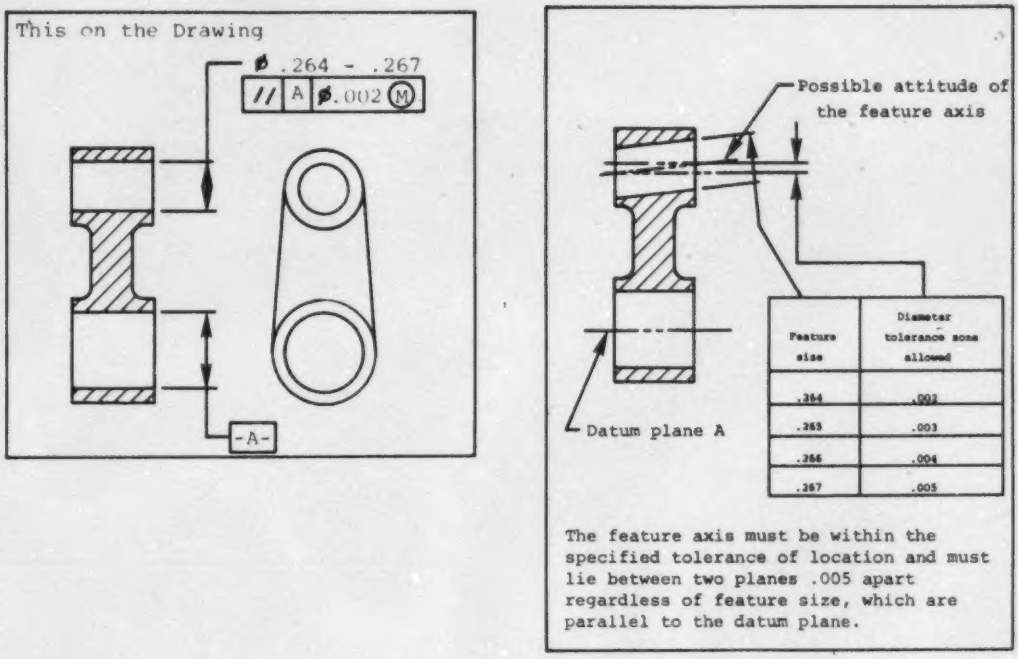


Figure 27-5-19 Parallelism Form Control with Diameter Feature Symbol (Rule #3)

Measurement of parallelism involves measuring the distance between the feature surface or axis and the datum surface or axis. Indicators or profile tracing equipment are most commonly used.

LOCATION TOLERANCES

Location tolerances control position, concentricity, and symmetry (See Figure 27-5-2). These tolerances involve related features and serve to locate features as required

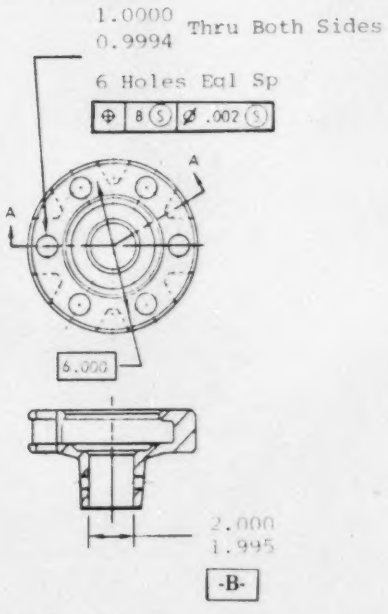
for assembly (fit) and function. RFS or MMC applies in all cases in accordance with Rule #2.

Position

A positional tolerance defines a zone within the axis or center plane of a feature is permitted to vary from true position (theoretically exact position). Positional tolerance is normally applied to internal mating features such as holes and slots and external mating features such as shafts, bosses, lugs and tabs.

The most restrictive application of positional tolerance is the RFS basis. When RFS applies, the feature's location must be within the positional tolerance from theoretically true position regardless of feature size. This concept is illustrated in Figure 27-5-20.

DRAWING



Section A-A

INTERPRETATION

1. Position of the holes must be basically 60° apart $(\frac{360^{\circ}}{6})$ on a 6.000 diameter and within a 0.002 cylindrical tolerance zone of such basic position as measured to datum B regardless of hole actual diameter or datum B actual diameter.
2. A hole at LMC is as accurately positioned as a hole at MMC.
3. The RFS application prohibits shift in hole position with changes in the diameter of datum B, i.e., the axis of a hole is positioned in relation to the axis of datum B.

Figure 27-5-20 Positional Tolerance on an RFS Basis

When positional tolerance applies at MMC, it can be interpreted in terms of the surface of the feature or axis of the feature. This is illustrated in Figure 27-5-21 by using a hole as the example.

Positional features are measured by comparative techniques (Module 27-3) and optics or autocollimator (Module 27-4).

Concentricity

Concentricity defines a cylindrical tolerance zone within which the axes of all cross-sectional elements of a feature's surface of revolution must lie with respect to the axis of a datum feature. As with all location tolerances, RFS or MMC is specified by the drawing in accordance with Rule #2. Concentricity is normally measured by comparative techniques (Module 27-3). A typical application is illustrated in Figure 27-5-22.

Symmetry

Symmetry defines two parallel planes separated by the symmetry tolerance and equally disposed about the center plane of the datum within which the center plane of the feature must lie. Symmetry is illustrated in Figure 27-5-23. It is measured with the same techniques as for position.

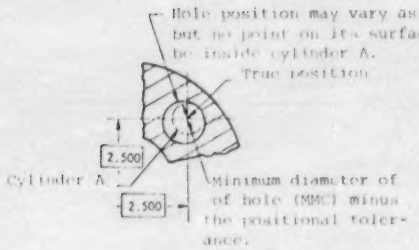
Runout

Runout is a composite tolerance used to control the functional relationship of features to a datum axis. Such features are those constructed around or at right angles to a datum axis.

⌖	⌀.xxx Ⓜ

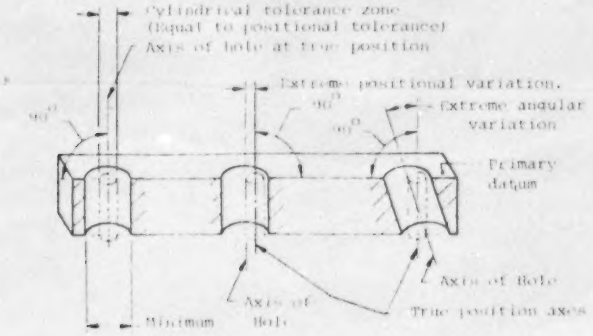
Means:

1. In terms of the cylindrical surface of the hole:


Hole position may vary as shown but no point on its surface shall be inside cylinder A.
True position
2.500
Cylinder A
2.500
Minimum diameter of of hole (MMC) minus the positional tolerance.

No element of the hole surface can be inside a theoretical cylinder, located at true position, having a diameter equal to that of Cylinder A (MCC feature diameter minus the positional tolerance).

2. In terms of the axis of the hole:


Cylindrical tolerance zone (Equal to positional tolerance)
Axis of hole at true position
Extreme positional variation.
90°
90°
90°
Extreme angular variation
Primary datum
Axis of Hole
Axis of Hole
True position axes
Minimum hole diameter
Axis of hole is coincident with true position axis
Axis of hole is located at extreme position to the left of true position axis (but within tolerance zone
Axis of hole is inclined to extreme position within tolerance zone
(a)
(b)
(c)

Note that the length of the tolerance zone is equal to the length of the feature, unless otherwise specified on the drawing.

The axis of the hole must fall within a cylindrical tolerance zone whose axis is located at true position and having a diameter equal to the positional tolerance. Such diameter increases as the feature diameter departs from MMC.

Figure 27-5-21 Positional Tolerance on an MMC Basis

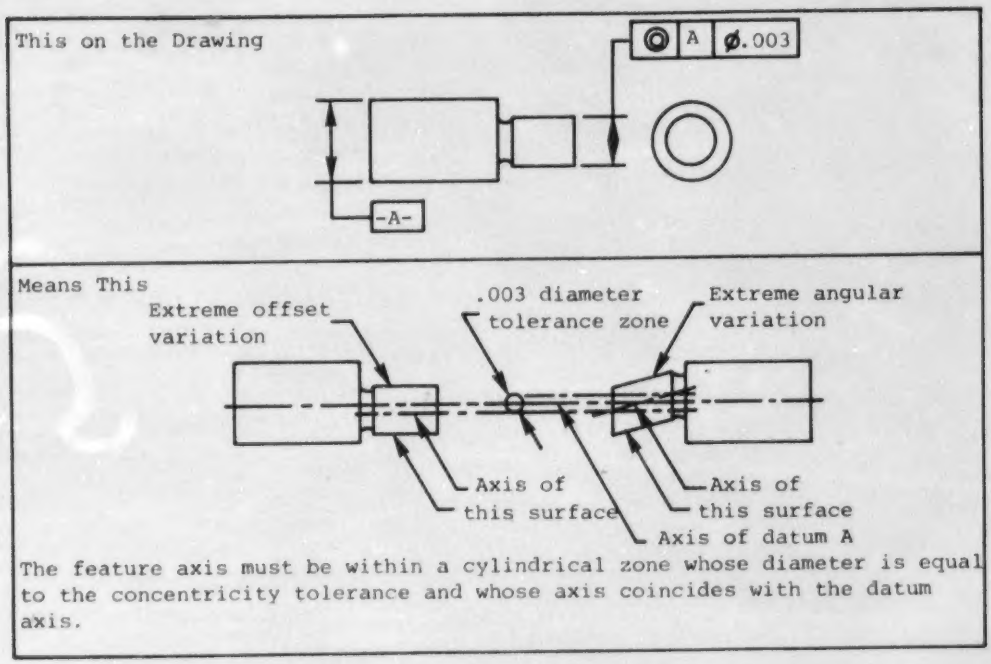

This on the Drawing
A
ø.003
-A-
Means This
Extreme offset variation
.003 diameter tolerance zone
Extreme angular variation
Axis of this surface
Axis of this surface
Axis of datum A
The feature axis must be within a cylindrical zone whose diameter is equal to the concentricity tolerance and whose axis coincides with the datum axis.

Figure 27-5-22 Concentricity Location Control (RFS)

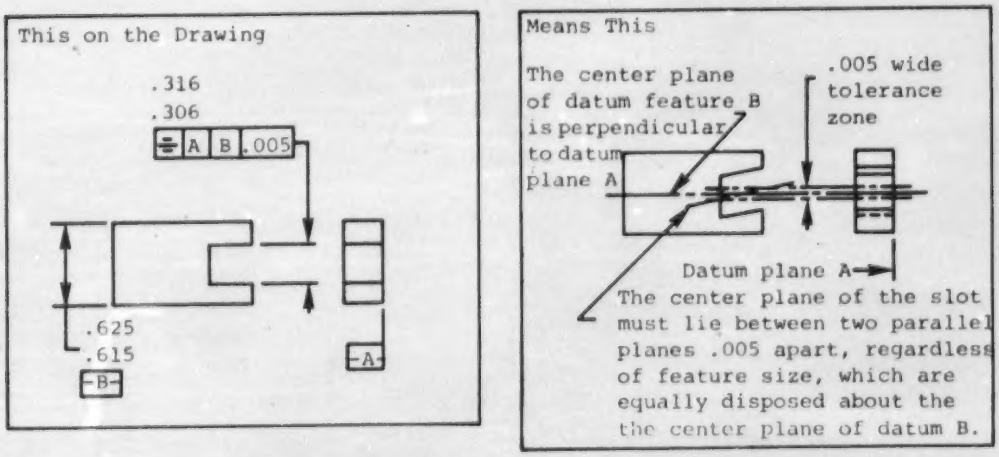
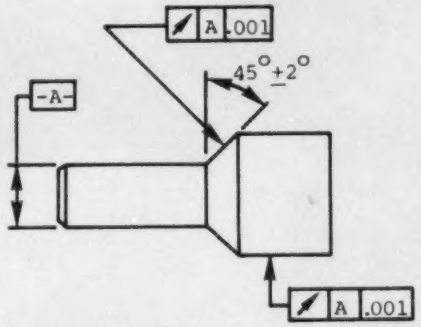

This on the Drawing
.316
.306
A
B
.005
.625
.615
-A-
-B-
Means This
The center plane of datum feature B is perpendicular to datum plane A
.005 wide tolerance zone
Datum plane A
The center plane of the slot must lie between two parallel planes .005 apart, regardless of feature size, which are equally disposed about the the center plane of datum B.

Figure 27-5-23 Symmetry Location Control (RFS)

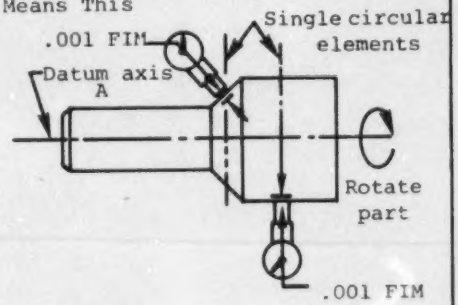
Runout is measured by comparative indicators (See Module 27-3), such that the maximum detected variation is compared with the runout tolerance to determine acceptance. Thus, "full indicator movement (FIM)" constitutes the measured value. (NOTE: prior texts may use such terms as full indicator reading (FIR) or total indicator reading (TIR); these former terms have identical meaning as FIM.

There are two types of runout control: circular and total. Where total applies the word TOTAL must be beneath the runout control symbol. Circular and total runout is illustrated in Figure 27-5-24 and Figure 27-5-25 respectively. As illustrated, Rule #1 applies to runout. (Please refer to the next page for Figures 27-5-24 and 27-5-25).

This on the Drawing



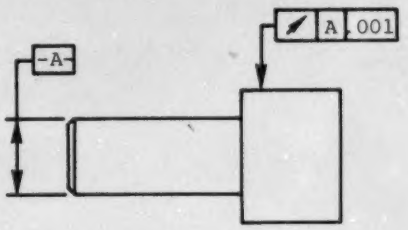
Means This



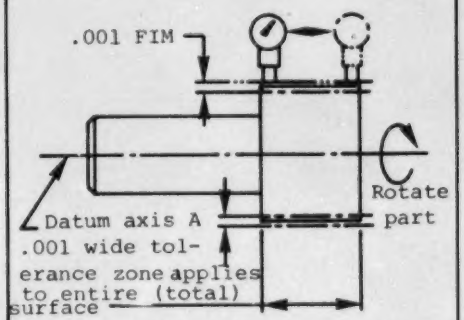
The features must be within the specified tolerance of size. At any measuring position, each circular element of these surfaces must be within the specified runout tolerance (.001 full indicator movement) when the part is rotated 360° about the datum-axis with the indicator fixed in a position normal to the true geometric surface. (This does not control the profile elements of these surfaces. Only the circular elements are controlled.) Whether the indicator is oriented normal to the actual surface or the true geometric (theoretically exact) surface will cause only a slight "cosine error" change in the magnitude of the FIM reading.

Figure 27-5-24 Circular Runout

This on the Drawing



Means This



The feature must be within the specified tolerance of size. The entire surface must lie within the specified runout tolerance zone (.001 full indicator movement) when the part is rotated 360° about the datum axis with the indicator placed at every location along the surface in a position normal to the true geometric surface, without reset of the indicator. (This controls the cumulative profile and circular elements of the entire surface). Whether the indicator is oriented normal to the actual surface or the true geometric (theoretically exact) surface will cause only a slight "cosine error" change in the magnitude of the FIM reading.

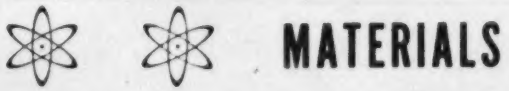
Figure 27-5-25 Total Runout

EXERCISES

Obtain engineering drawings for mechanial components or systems and using these drawings as a reference discuss or answer the following:

1. Are the dimensions recorded on the drawings given to the same number of decimal places as the actual physical measurement? Briefly discuss your answer.
2. List and briefly discuss four tolerancing methods. Are there examples of these on your drawings? If so give an example of each tolerancing method.
3. Show how the dimension 1.6526 inches could be rounded off to:
   a. four decimals
   b. three decimals
   c. two decimals
   d. one decimal
   e. a whole number
4. Discuss the applicable rule for the case which applies to drawings giving size limits but not form or location controls.
5. Discuss the rule which applies to the form controls which are applicable to the axis of the feature.
6. Discuss the rule applying to a drawing giving location controls.
7. Discuss the rule(s) applying to the following form tolerances:
   a. Straightness
   b. Flatness
   c. Roundness
   d. Cylindricity
   e. Angularity
   f. Perpendicularity
   g. Parallelism

8. Discuss the rule(s) applying to the following location tolerances:
   a. Position
   b. Concentricity
   c. Symmetry
9. Runout is a composite tolerance applicable to related features. Discuss the rule which applies to this tolerance.
10. To reinforce the dimensional tolerances discussed in this module, perform measurements for parallelism, angularity and flatness described in the Procedures section.



MATERIALS

1. Test indicator
2. Weight stand
3. Surface plate
4. Sine bar
5. Autocollimator and optical targets
6. Gage blocks
7. Parallel bar

PROCEDURES

1. Measure the parallelism of either a gage block or parallel bar using a test indicator, height stand and surface plate.
2. Measure angularity of arc angle block using a sine bar and test indicator.
3. Measure the flatness of a surface plate using an autocollimator and optical targets.

DATA TABLE

None.

REFERENCE MATERIALS

ANSI Y14.5-1973, American Society of Mechanical Engineers, Dimensioning and Tolerancing, American Society of Mechanical Engineers, 1973.

Society of Manufacturing Engineers (SME), Handbook of Industrial Metrology, Prentice-Hall, Inc., 1967.

Farago, Francis T., PhD, Handbook of Dimensional Measurement, Industrial Press Inc., Third Printing, 1968.

Technical Committee ISO/TC 12, ISO 1000, American National Standards Institute, 1973.

TEST

Read each questions carefully. Be sure to provide the specific information each question requires. Make your answer brief. Use the backs of the test if you need more space.

1. Define the following terms:

a. maximum material condition _____

b. least material condition _____

c. regardless of feature size _____

d. flatness _____

e. cylindricity _____

f. parallelism _____

g. true position _____

h. concentricity _____

_____

_____

_____

2. Identify the following tolerance as limit, unilaterial or bilaterial:

a. 5.248 - 5.252

b. $1.125 \begin{matrix} +0.000 \\ -0.005 \end{matrix}$

c. 125.72 ± 0.05

3. Round the following to the specified number of decimal places:

a. 12.1458 to one decimal place

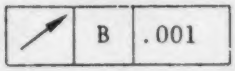
b. 1.5552 to two decimal places

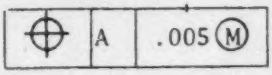
c. 3.7445 to three decimal places

4. - 15. Match the geometric characteristic with its symbol by placing the letter of the characteristic in the right hand column adjacent to the symbol in the left hand column:

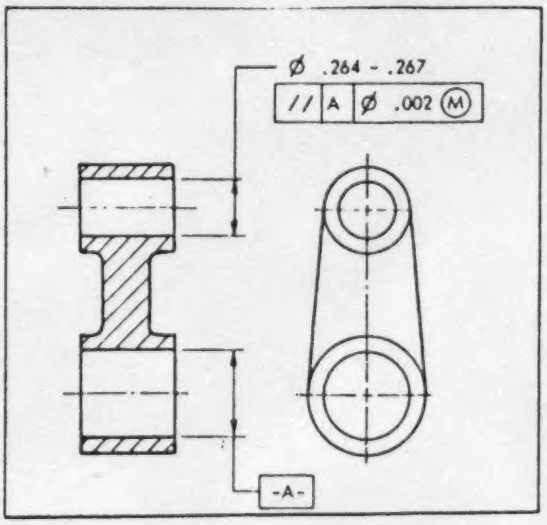
_____4.	—	(a) angularity
_____5.	▱	(b) concentricity
_____6.	○	(c) cylindricity
_____7.	⟋	(d) flatness
_____8.	⌭	(e) parallelism
_____9.	≡	(f) perpendicularity
_____10.	⌒	(g) position
_____11.	⌾	(h) profile of a line
_____12.	∠	(i) roundness
_____13.	⌖	(j) runout
_____14.	⊥	(k) straightness
_____15.	//	(l) symmetry

16. State the meaning of the following feature control symbols:

a. 

b. 

Use the following sketch in answering questions 17 and 18.



17. If the feature size in the above sketch is .264 inch what is the tolerance on the geometrical relationship of the feature's axis with datum A?

_____

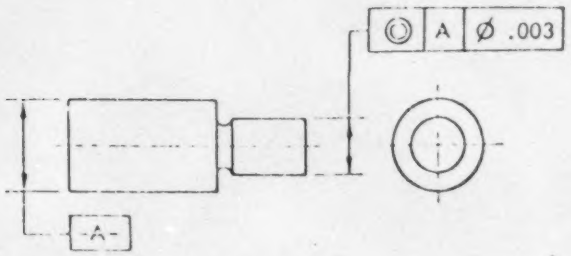
_____

18. If the feature size in the above sketch is .266 what is the tolerance on the geometrical relationship of the feature's axis with datum A?

_____

_____

19. If the feature size in the following sketch is .502, what is the tolerance of the geometrical relationship of the axis of the feature with datum A?



20. When a perpendicularity tolerance is specified for an axis, Rule #3 applies. Describe how an inspector would know from the feature control symbol if such a tolerance applies on an RFS or MMC basis.

_____

_____

_____

_____

21. When a position tolerance is specified for a feature, Rule #2 applies. Describe how an inspector would know from the feature control symbol if such a tolerance applies on an RFS or MMC basis.

APPENDIX A

27-5-53

APPENDIX A
GLOSSARY

This glossary is a quick reference for terms and definitions and should be referred to whenever new terms are encountered.

- Actual Size - An actual size is the measured size of the feature.
- Angularity - Angularity is the condition of a surface, axis, or center plane which is at a specified angle (other than 90°) from a datum plane or axis. Symbol: ∠
- Basic Dimension - A dimensional specified on a drawing as BASIC (or abbreviated BSC) is a theoretical value used to describe the exact size, shape, or location of a feature. It is used as the basis from which permissible variations are established by tolerances on other dimensions or notes. A basic dimension can be symbolized by boxing it: ☐
- Basic Size - The basic size is that size from which limits of size are derived by the application of allowances and tolerances.
- Bilaterial Tolerancing - A bilaterial tolerance is a tolerance in which variation is permitted in both directions from the specified dimension.
- Center Plane - Center plane is the middle or median plane of a feature.
- Circularity - See Roundness.
- Concentricity - Concentricity is a condition in which two or more features (cylinders, cones, spheres, hexagons, etc.) in any combination have a common axis. Symbol: ◎

- Cylindricity - Cylindricity is a condition of a surface of revolution in which all points of the surface are equidistant from a common axis. Symbol: ⌭

- Datum - Datums are points, lines, planes, cylinders, axes, etc., assumed to be exact for purposes of computation, from which the location or geometric relationship (form) of features of a part may be established.
- Datum Identification Symbol - The datum identification symbol contains the datum reference letter in a drawn rectangular box, e.g., [-C-]
- Datum Target - A datum target is a specified datum point, line, or area identified on the drawing with a datum target symbol ⊕ used to establish datum points, lines, planes, or areas of special function, or manufacturing and inspection repeatability.
- Dimension - A dimension is a numerical value expressed in appropriate units of measure and indicated on a drawing along with lines, symbols, and notes to define the geometrical characteristics of an object.
- Feature - A feature is component portion of a part that can be used as the basis for a datum.
- Feature Control Symbol - The feature control symbol is a rectangular box containing the geometric characteristics symbol and the form or location tolerance. If necessary, datum references and modifiers applicable to the feature or the datums are also contained in the box e.g., [// A .005]
- Fit - Fit is the general term used to signify the range of the tightness or looseness which may result from the application of a specific combination of

allowances and tolerance in the design of mating part features. Fits are of four general types: clearance, interference, transition, and line.

- Flatness - Flatness is the condition of a surface having all elements in one plane. Symbol: ▱
- Form Tolerance - A form tolerance states how far actual surfaces or features are permitted to vary from the desired form implied by the drawing. Expressions of these tolerances refer to flatness, straightness, parallelism, perpendicularity, angularity, roundness, cylindricity, runout, profile of any surface, and profile of any line.
- Full Indicator Movement (FIM) - Full indicator movement is the total indicator movement observed with the dial indicator in contact with the part feature surface during one full revolution of the part about its datum axis. Full indicator movement also refers to tne total indicator movement observed while in traverse over a fixed noncircular shape.
- Geometric Characteristics - Geometric characteristics refer to the basic elements or building blocks which form the language of geometric dimensioning and tolerancing. Generally, it refers to the symbols used in both form and location tolerancing.
- Least Material Condition - The condition where a feature of size contains the least amount of material within the stated limits of size, e.g., largest hole size and smallest shaft size.
- Limits of Size - The limits of size are the applicable maximum and minimum sizes of a feature.
- Location Tolerance - A location tolerance states how far individual features may vary from the perfect location implied by the drawing. Expressions of

these tolerances normally refer to position, concentricity, and symmetry.

- Maximum Material Condition (MMC) - The condition where a feature size contains the maximum amount of material within the stated limits of size, e.g., minimum hole size and a maximum shaft size. Symbol: Ⓜ
- Median Plane - Median plane is the middle or center plane of a feature.
- Minimum Material Condition - See Least Material Condition.
- Modifier - A modifier is the term used to describe the application of the maximum material condition or "regardless of feature size" principles. The modifiers are maximum material condition (MMC). symbol Ⓜ , and regardless of feature size (RFS), symbol Ⓢ .
- Normality - See Perpendicularity.
- Parallelism - Parallelism is the condition of a surface, line, or axis which is equidistant at all points from a datum plane or axis. Symbol: //
- Perpendicularity - Perpendicularity is the condition of a surface, axis, or line which is 90° from a datum plane or a datum axis. Symbol: ⊥
- Positional Tolerance - The permissible variation of a feature from the true position. Symbol: ⌖
- Profile of Any Line - Profile of any line is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterally, along a line element of a feature. Symbol: ⌒
- Profile of Any Surface - Profile of any surface is the condition permitting a uniform amount of profile variation, either unilaterally or bilaterlaly, on a surface. Symbol: ⌓

- Projected Tolerance Zone - A projected tolerance zone is a tolerance zone applied to a hole in which a pin, stud, screw, or bolt, etc., is to be inserted. It controls the perpendicularity of the hole to the extent of the projection from the hole and as it relates to the mating part clearnace. The projected tolerance zone extends above the surface of the part to the functional length of the pin, screw, etc., relative to its assembly with the mating part. Symbol: (P)
- Regardless of Feature Size (RFS) - This is the condition where the tolerance of form or position must be met irrespective of where the feature lies within its size tolerance. As applied to datums, it refers to the actual size of the datum feature. Symbol: (S)
- Roundness - Roundness is the condition on a surface of revolution (cylinder, cone, sphere) where all points of the surface intersected by any plane (1) perpendicular to a common axis (cylinder, cone) or (2) passing through a common center (sphere) are equidistant from the axis. Symbol: ◯
- Runout - Runout is the deviation from the desired form of a part surface of revolution detected during full rotation of the part on a datum axis when using a dial indicator (or equivalent measuring device). Symbol: ↗ Circular Runout: ⌀ Total Runout
- Size Tolerance - A size tolerance states how far individual features may vary from the desired size. Size tolerances are specified with either unilateral, bilateral, or limit tolerancing methods.
- Squareness - See Perpendicularity.

- Straightness - Straightness is a condition where an element of a surface is a straight line. Symbol: ⏤

- Symmetry - Symmetry is a condition in which a feature is symmetrically disposed about the center plane of a datum feature. Symbol: ⌯
- Tolerance - A tolerance is the total amount by which a specific dimension may vary; thus, the tolerance is the difference between limits.
- True Position - True position is a term used to describe the theoretically perfect (exact) location of a point, line, or plane (normally the center) of a feature in relationship with a datum reference or other feature.
- Virtual Size - Virtual size of a feature is the effective size of the profile that must be considered in determining the clearance between mating parts or features. It is a derived size generated from the collective effect of all profile variations permitted by the specified tolerances. It represents the most extreme condition of assembly at MMC.